CFD ANALYSIS OF HEAT TRANSFER IN A HELICAL COIL HEAT EXCHANGER USING FLUENT

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

BACHELOR IN TECHNOLOGY
IN
MECHANICAL ENGINEERING

Department of Mechanical Engineering

National Institute of Technology, Rourkela

Rourkela 769008

CFD ANALYSIS OF HEAT TRANSFER IN A HELICAL COIL HEAT EXCHANGER USING FLUENT

A thesis submitted in partial fulfillment of the requirements for the degree of

Bachelor of technology

In

Mechanical engineering

By

Siddhartha Shankar Behera (109ME0258)

Under the guidance of

Dr. A. K. Satapathy

Department of Mechanical Engineering

National Institute of Technology

Rourkela 769008

C E R T I F I C A T E

This is to certify that the work in this thesis entitled "CFD analysis of heat transfer in a helical coil heat exchanger using fluent" by ***Siddhartha Shankar Behera***, has been carried out under my supervision in partial fulfillment of the requirements for the degree of *Bachelor of Technology in Mechanical Engineering* during session 2012-2013 in the *Department of Mechanical Engineering, National Institute of Technology, Rourkela*.

To the best of my knowledge, this work has not been submitted to any other University/Institute for the award of any degree or diploma.

Date: 07/05/2013

Dr. Ashok Kumar Satapathy

(Supervisor)

Associate Professor

Department of Mechanical Engineering

National Institute of Technology, Rourkela

ACKNOWLEDGEMENT

I am extremely fortunate to be involved in an exciting and challenging research project like "CFD analysis of heat transfer in a helical coil heat exchanger using fluent". It has enriched my life, giving me an opportunity to work in a new environment of Fluent. This project increased my thinking and understanding capability as I started the project from scratch. I would like to express my greatest gratitude and respect to my Supervisor ***Dr. Ashok Kumar Satapathy***, for his excellent guidance, valuable suggestions and endless support. He has not only been a wonderful supervisor but also a genuine person. I consider myself extremely lucky to be able to work under the guidance of such a dynamic personality. Actually he is one of such genuine person for whom my words will not be enough to express.

I am also thankful to ***Dr. K. P. Maity*, H.O.D** of Department of Mechanical Engineering, National Institute of Technology, Rourkela for his constant support and encouragement.

Last, but not the least I extend my sincere thanks to all faculty members of Mechanical Engineering department for making my project a successful one, for their valuable advice in every stage and also giving me absolute working environment where I unlashed my potential. I would like to thank all whose direct and indirect support helped me completing my thesis in time. I want to convey my heartiest gratitude to my parents for their unfathomable encouragement.

Siddhartha Shankar Behera

109ME0258

Bachelor of Technology, Mechanical Engineering

CONTENTS

Certificate i

Acknowledgement ii

List of Tables iv

List of Figures v

Abstract vi

Nomenclature vii

1. Introduction About the Project 1
   - 1.1 Tubular Heat Exchangers 3
   - 1.2 Double Pipe Heat Exchanger 3
   - 1.3 Heat Transfer Coefficients 4
   - 1.4 Advantages and Disadvantages 5
   - 1.5 Applications 6
   - 1.6 Aim of Present Work 6
   - 1.7 Introduction to CFD 7
   - 1.8 Organization of Thesis 8
2. Literature Survey 9
3. CFD Analysis 12
   - 3.1 Geometry 13
     - 3.1.1 Sketching 13
     - 3.1.2 Sweep 13
     - 3.1.3 Merging 13
   - 3.2 Mesh 15
     - 3.2.1 y^+ Values 15
     - 3.2.2 Named Selection 16
   - 3.3 Solution 17

3.3.1 Problem Setup 17

3.3.2 Models 17

3.3.3 Materials 17

3.3.4 Cell Zone Conditions 17

3.3.5 Boundary Conditions 18

3.3.6 Reference Values 18

3.3.7 Solution Methods 18

3.3.8 Solution Controls and Initialization 19

3.3.9 Measure of Convergence 19

3.3.10 Run Calculation 20

4. Results and Discussions 21

4.1 Mass Flow Rate and Total Heat Transfer Rate 22

4.2 Contours 23

4.3 Vectors 27

4.4 Plots 29

5. Conclusions 31

6. References 32

List of Tables

Figure Number	Caption	Page Number
1	Naming Of Various Parts of The Body	14
2	y^+ Values for Different Wall Treatments	15
3	Boundary Conditions	18
4	Residuals	19
5	Mass Flow Rate	22
6	Total Heat Transfer Rate	22

List of Figures

Figure Number	Caption	Page Number
1 (A)	Double Pipe Helical Coil	3
1 (B)	Close-Up of Double Pipe Coil	3
2	Wall Convection	4
3	Original Geometry	14
4	Geometry for Meshing	14
5	Mesh	16
6	Close-Up View of The Mesh	16
7	Named Selections	17
8	Contours of Dynamic Pressure In Pascal	23
9	Contours of Total Pressure In Pascal	23
10	Contours of Static Temperature	24
11	Contours of Turbulent Dissipation Rate	24
12	Contours of Turbulent Kinetic Energy	25
13	Contours of Effective Thermal Conductivity	25
14	Contours of Total Surface Heat Flux	26
15	Contours of Cell Reynolds Number	26
16	Contours of Effective Prandtl Number	27
17	Velocity Vectors Colored by Velocity Magnitude	27
18	Velocity Vectors Colored by Velocity Magnitude - Close Up	28
19	Velocity Vectors Colored by Static Pressure	28
20	Scaled Reisduals	29
21	Wall Shear Stress Plot	29
22	Total Pressure Plot	30
23	Surface Nusselt Number	30

ABSTRACT

Heat exchangers are the important engineering systems with wide variety of applications including power plants, nuclear reactors, refrigeration and air-conditioning systems, heat recovery systems, chemical processing and food industries. Helical coil configuration is very effective for heat exchangers and chemical reactors because they can accommodate a large heat transfer area in a small space, with high heat transfer coefficients. This project deals with the analysis of the helical coiled heat exchanger with various correlations given by different papers for specific conditions. Although various configurations are available, the basic and most common design consists of a series of stacked helically coiled tubes placed in a cylindrical outer cover. The inner tube ends are connected to manifolds, which act as fluid entry and exit locations. And the outer tube is also provided with inlet and outlet manifolds so that cooling fluid can be passed through it. The tube bundle is constructed of a number of tubes stacked atop each other, and the entire bundle is placed inside a helical casing, or shell. The complex fluid-dynamic inside curved pipe heat exchangers gives them important advantages over the performance of straight tubes in terms of area/volume ratio and enhancing of heat transfer and mass transfer coefficient. Convective heat transfer between a surface and the surrounding fluid in a heat exchanger has been a major issue and a topic of study for a long time. The analysis of these various correlations with certain defined data is presented in this project. In this study, an attempt has been made to analyze the effect of counter-flow on the total heat transfer from a helical tube. The temperature contours, velocity vectors, surface nusselt number, total heat transfer rate from the wall of the tube was calculated and plotted using ANSYS 13.0. Copper was chosen as the metal for the construction of the helical tube. The fluid flowing through the inner tube and outer casing was taken as water.

NOMENCLATURE

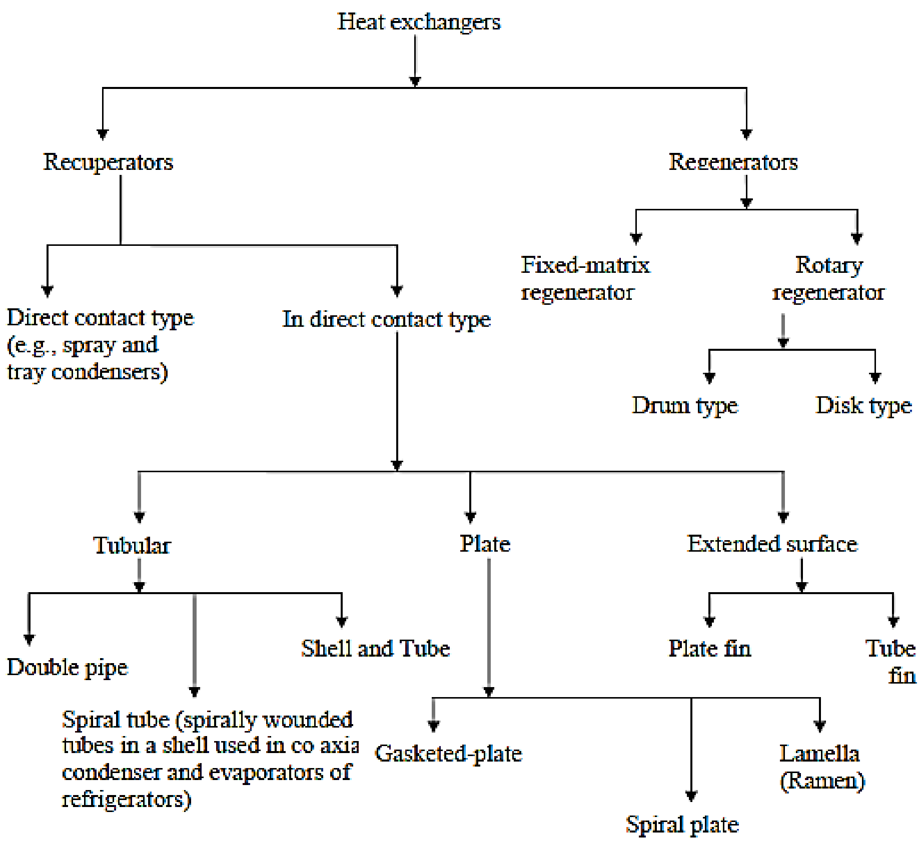
- A = area of heat transfer (m^2)
- De= Dean Number
- H = heat transfer coefficient (Wm^{-2} K^{-1})
- H = tube pitch (m)
- K = thermal conductivity ($Wm^{-1}K^{-1}$)
- L= length of the pipe (m)
- Nu= Nusselt number
- Pr = Prandtl number
- Q = heat transferred (W)
- R= inner radius of the tube (m)
- R = resistance the flow of thermal energy ($W^{-1}m^2K$)
- Rc = pitch circle radius of the pipe (m)
- Re= Reynolds number
- U = velocity (ms^{-1})
- U = overall heat transfer coefficient ($Wm^{-2}K^{-1}$)
- V= volume (m^3)
- A = helix angle (rad)
- δ = curvature ratio
- Δ = (temperature) difference (K)
- μ = viscosity ($kgm^{-1}s^{-1}$)
- ρ = density (kgm^{-3})
- av = average
- i = internal
- LM = log mean
- o = external
- ov = overall
- w = wall

CHAPTER 1

INTRODUCTION ABOUT THE PROJECT

Heat exchange between flowing fluids is one of the most important physical process of concern, and a variety of heat exchangers are used in different type of installations, as in process industries, compact heat exchangers nuclear power plant, HVACs, food processing, refrigeration, etc. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. The heat transfer occurs by three principles: conduction, convection and radiation. In a heat exchanger the heat transfer through radiation is not taken into account as it is negligible in comparison to conduction and convection. Conduction takes place when the heat from the high temperature fluid flows through the surrounding solid wall. The conductive heat transfer can be maximized by selecting a minimum thickness of wall of a highly conductive material. But convection is plays the major role in the performance of a heat exchanger.

Forced convection in a heat exchanger transfers the heat from one moving stream to another stream through the wall of the pipe. The cooler fluid removes heat from the hotter fluid as it flows along or across it.



Classification of Heat Exchanger

1.1 Tubular Heat Exchangers:

Tubular heat exchangers are built of mainly of circular tubes although some other geometry has also been used in different applications. This type of construction offers a large amount of flexibility in design as the designing parameters like the diameter, length and the arrangement can be easily modified. This type is used for liquid-to-liquid (phase changing like condensing or evaporation) heat transfer. Again this type is classified into shell and tube, double pipe and spiral tube heat exchangers.

1.2 Double Pipe Heat Exchanger:

The double pipe or the tube in tube type heat exchanger consists of one pipe placed concentrically inside another pipe having a greater diameter. The flow in this configuration can be of two types: parallel flow and counter-flow. It can be arranged in a lot of series and parallel configurations to meet the different heat transfer requirements. Of this the helically arranged stands out as it has found its place in different industrial applications. As this configuration is widely used, knowledge about the heat transfer coefficient, pressure drop, and different flow patterns has been of much importance. The curvature in the tubes creates a secondary flow, which is normal to the primary axial direction of flow. This secondary flow increases the heat transfer between the wall and the flowing fluid. And they offer a greater heat transfer area within a small space, with greater heat transfer coefficients. Study has been done on the types of flows in the curved pipes, and the effect of Prandtl and Reynolds number on the flow patterns and on Nusselt numbers. The two basic boundary conditions that are faced in the applications are constant temperature and the constant heat flux of the wall.

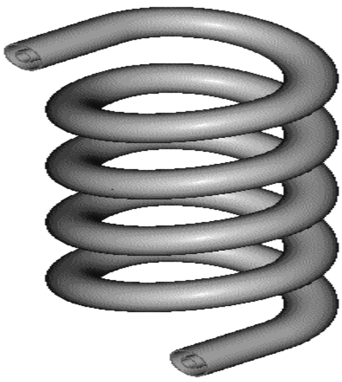


Fig.1(a) Double pipe helical coil

Fig.1(b) Close-up of double pipe coil

1.3 Heat Transfer Coefficient:

Convective heat transfer is the transfer of heat from one place to another by the movement of fluids due to the difference in density across a film of the surrounding fluid over the hot surface. Through this film heat transfer takes place by thermal conduction and as thermal conductivity of most fluids is low, the main resistance lies there. Heat transfer through the film can be enhanced by increasing the velocity of the fluid flowing over the surface which results in reduction in thickness of film. The equation for rate of heat transfer by convection under steady state is given by,

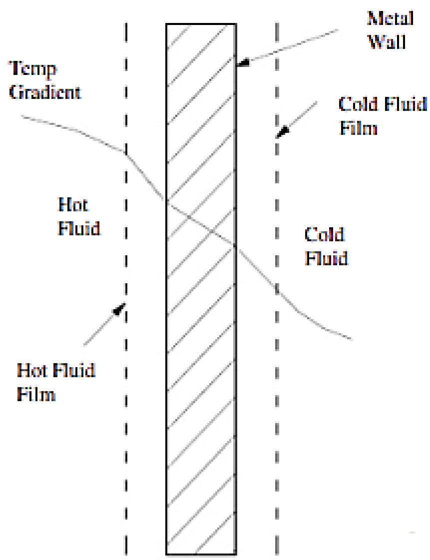


Figure 2 Wall Convection

$Q = h\ A\ (T_w - T_{atm})$

Where, h-is the film coefficient or surface coefficient ($W/m^2.K$).

A is the area of the wall,

T_w is the wall temperature,

T_{atm} is surrounding temperature.

The value of 'h' depends upon the properties of fluid within the film region; hence it is called 'Heat Transfer Coefficient'. It depends on various properties of fluid, linear dimensions of surface and fluid velocity (i.e. nature of flow).

The overall heat transfer coefficient is the overall transfer rate of a series or parallel combination of convective and conductive walls. The 'overall Heat Transfer Coefficient' is expressed in terms of thermal resistances of each fluid stream. The summation of individual resistances is the total thermal resistance and its inverse is the overall heat transfer coefficient, U.

$$\frac{1}{U} = \frac{1}{h_o} + \frac{A_o}{A_i}\frac{1}{h_i} + R_{fo} + \frac{A_o}{A_i}R_{fi} + R_w$$

Where, U = overall heat transfer coefficient based on outside area of tube all

A = area of tube wall

h = convective heat transfer coefficient

R_f = thermal resistance due to fouling

R_w= thermal resistance due to wall conduction and suffixes 'I' and 'O' refer to the inner and outer tubes, respectively.

Due to existence of the secondary flow, the heat transfer rates (& the fluid pressure drop) are greater in the case of a curved tube than in a corresponding straight tube at the same flow rate and the same temperature and same boundary conditions.

1.4 Advantages & Disadvantages:

1.4.1 Advantages of coils:

1) Coils give better heat transfer performance, since they have lower wall resistance & higher process side coefficient.
2) The whole surface area of the curved pipe is exposed to the moving fluid, which eliminates the dead-zones that are a common drawback in the shell and tube type heat exchanger.
3) A coil can provide a large surface area in a relatively small reactor volume.
4) The heat exchanger's spring-like coil eliminates thermal expansion and thermal shock problems, which helps in high pressure operations.
5) Fouling is comparatively less in helical coil type than shell and tube type because of greater turbulence created inside the curved pipes.

1.4.2 Disadvantages of coils:

1) For highly reactive fluids or highly corrosive fluid coils cannot be used, instead jackets are used.
2) Cleaning of vessels with coils is more difficult than the cleaning of shells and jackets.
3) Coils play a major role in selection of agitation system. Densely packed coils can create unmixed regions by interfering with fluid flow.
4) The design of the helical tube in tube type heat exchanger is also a bit complex and challenging.

1.5 Applications:

Use of helical coils for heat transfer applications:

1) Used in hydro carbon processing, recovery of CO_2, cooling of liquid hydrocarbons, also used in polymer industries for cooling purposes.
2) Helical coils are used for transferring heat in chemical reactors because the heat transfer coefficients are greater in helical coils as compared to other configurations. This is especially important when chemical reactions have high heats of reaction are carried out and the heat consumed has to be transferred rapidly to maintain the temperature of the reaction. They are used widely in petroleum industries for different applications.
3) Because of the compact configuration of helical coils, they can be readily used in heat transfer application with space limitations, for example, marine cooling systems, central cooling, cooling of lubrication oil, steam generations in marine and industrial applications.
4) The helical coiled heat exchangers are used widely in food and beverage industries, like in food processing and pre-heating, pasteurization of liquid food items, and for storing them at desired temperatures.
5) Helical coil heat exchangers are often used as condensers in used in HVACs due to their greater heat transfer rate and compact structure.
6) Helical coiled tubes have been and are used extensively in cryogenic industry for the liquefaction of gases.

1.6 Aim of the Present Work:

The design of a helical coil tube in tube heat exchanger has been facing problems because of the lack of experimental data available regarding the behavior of the fluid in helical coils and also in case of heat transfer data, which is not the case in Shell & Tube Heat Exchanger. So to the best of our effort, numerical analysis was carried out to determine the heat transfer characteristics for a double-pipe helical heat exchanger by varying the different parameters like different temperatures and diameters of pipe and coil and also to determine the fluid flow pattern in helical coiled heat exchanger. The objective of the project is to obtain a better and more quantitative insight into the heat transfer process that occurs when a fluid flows in a helically coiled tube. The study also covered the different types of fluid flow range extending from laminar flow through transition to turbulent flow. The materials for the study were decided and fluid taken was water and the material for the pipe was taken to be copper for its better conducting properties.

1.7 Introduction to CFD:

Computational Fluid Dynamics, abbreviated as CFD, uses different numerical methods and a number of computerized algorithms in order to solve and analyze problems that involve the flow of fluids. The calculations required simulating the interaction of fluids with surfaces defined by boundary conditions, and initial conditions are done by the ANSYS Fluent v13.0. The Navier-Stokes equations form the basis of all CFD problems. Two equation models are used for the simulations, and different models are discussed below.

The continuity equation, energy equation and the Navier-Stokes momentum equation govern the flow of the fluid in the curve tubes

Continuity Equation gives the conservation of mass and is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_1}{\partial x_1} + \frac{\partial \rho U_2}{\partial x_2} + \frac{\partial \rho U_3}{\partial x_3} = 0$$

$$=> \frac{\partial U}{\partial x} + \frac{\partial V}{\partial x} = 0$$

And for constant density, $\frac{\partial \rho}{\partial t} = 0$

The momentum balance, (Navier-Stokes equations) follows Newton's 2nd law. The two forces acting on the finite element are the body and the surface forces. In CFD programs, the momentum equation is given as

$$\rho \left(u \frac{\partial U}{\partial x} + v \frac{\partial V}{\partial x} \right) = -\rho g - \frac{\partial p}{\partial x} + \mu \frac{\partial 2y}{\partial x2}$$

The governing energy equation is: $\rho C_p \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = k \frac{\partial^2 T}{\partial y^2}$

Turbulence is created because of the unstable nature of the fluid flow. The flow becomes turbulent for higher Reynolds number. In this model the k-ε (turbulent kinetics energy "k" and the turbulent dissipation "ε") model is used.

The physical interpretation of the ε equation is,

1. Accumulation of ε
2. Convection of ε by the mean velocity
3. Production of ε
4. Dissipation of ε
5. Diffusion of ε

The time constant for turbulence is calculated from the turbulent kinetic energy and dissipation rate of turbulent kinetic energy.

$$\tau = \frac{k}{\varepsilon}$$

1.8 Organization of Thesis:

The entire thesis is divided into 4 chapters:

First Chapter is all about the introduction about the project. It consists of the pre-ideas related to the project. It includes the objective and the work division of the thesis.

Second Chapter consists of the literature survey done during the project work. It includes the briefs of the important references from journals and papers that were referred in course of the project.

Third Chapter tells the reader about the fundamentals of the CFD analysis. The process of building geometry, meshing the geometric model, giving suitable cell-zone and boundary conditions and solving the problem. It gives the complete description of the work. It involves the methodology and the various steps taken to obtain the desired results.

Fourth Chapter lists all the results and contours-plots-vectors obtained after the solving of the problem. The later parts list the conclusion, references used and the nomenclature used in the thesis.

CHAPTER 2

LITERATURE REVIEW

Daniel Flórez-Orrego, Walter Arias, Diego López and Héctor Velásquez have worked on the single phase cone shaped helical coil heat exchanger. The study showed the flow and the heat transfer in the heat exchanger. An empirical correlation was proposed from the experimental data for the average nusselt number and a deviation of 23% was found. For the cone shaped helical coils an appreciable inclination of the velocity vector components in the secondary flow was seen, even though the contours of velocity were similar. The study showed that some of the deviations and errors were due to the non-uniform flame radiation and condensed combustion products which modified the conditions of the constant wall heat flux assumptions. The correlations for the nusselt number values were not totally reliable. There was no proper data available for the effect of the taper in the local nusselt number and also the effect of curvature ratio, vertical position and the pitch of the heat exchanger. [1]

J.S. Jayakumar observed that the use of constant values for the transfer and thermal properties of the fluid resulted in inaccurate heat transfer coefficients. Based on the CFD analysis results a correlation was developed in order to evaluate the heat transfer coefficient of the coil. In this study, analysis was done for both the constant wall temperature and constant wall heat flux boundary conditions. The nusselt numbers that were obtained were found to be highest on the outer coil and lowest in the inner side. Various numerical analyses were done so as to relate the coil parameters to heat transfer. The coil parameters like the diameters of the pipes, the Pitch Circle Diameters have significant effect on the heat transfer and the effect of the pitch is negligible. [2]

Timothy J. Rennie studied the heat transfer characteristics of a double pipe helical heat exchanger for both counter and parallel flow. Both the boundary conditions of constant heat flux and constant wall temperature were taken. The study showed that the results from the simulations were within the range of the pre-obtained results. For dean numbers ranging from 38 to 350 the overall heat transfer coefficients were determined. The results showed that the overall heat transfer coefficients varied directly with the inner dean number but the fluid flow conditions in the outer pipe had a major contribution on the overall heat transfer coefficient. The study showed that during the design of a double pipe helical heat exchanger the design of the outré pipe should get the highest priority in order to get a higher overall heat transfer coefficient. [3]

J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, Rohidas Bhoi studied the constant thermal and transport properties of the heat transfer medium and their effect on the prediction of heat transfer coefficients. Arbitrary boundary conditions were not applicable for the determination of heat transfer for a fluid-to-fluid heat exchanger. An experimental setup was made for studying the heat transfer and also CFD was used for the simulation of the heat transfer. The CFD simulation results were reasonably well within the range of the experimental results. Based on both the experimental and simulation results a correlation was established for the inner heat transfer coefficient. [4]

Usman Ur Rehman studied the heat transfer and flow distribution in a shell and tube heat exchanger and compared them with the experimental results. The model showed an average error of around 20% in the heat transfer and the pressure difference. The study showed that the symmetry of the plane assumption worked well for the length of the heat exchanger but not in the outlet and inlet regions. The model could be improved by using Reynold Stress models instead of k-ε models. The heat transfer was found to be on the lower side as there was not much interaction between the fluids. The design could be improved by improving the cross flow regions instead of the parallel flow. [5]

Nawras H. Mostafa, Qusay R. Al-hagag studied on the mechanical and thermal performance of elliptical tubes used for polymer heat exchangers. The mechanical analysis showed that the streamlined shape of the outer tube had an optimal thermal performance. A set of design curves were generated from which a number of geometries of the tube and different materials can be easily selected in order to meet the deformation constraints. A finite element solution was determined for strain as a function of the material of the tube. [6]

CHAPTER 3

CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modeling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modeling starts with the describing of the boundary and initial conditions for the dominion and leads to modeling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

3.1 Geometry:

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeler opens as a new window as the geometry is double clicked.

3.1.1 Sketching

Out of 3 planes, viz, XY-plane, YZ-plane and ZX-plane, the YZ-plane is selected for the first sketch. A 4 inch line for the height of the helical structure is made. A new plane is created in reference with the YZ-plane which is termed as plane 4. 4 new sketchers are added under the new plane, i.e. plane 4. In sketch 2, a circle of diameter 0.545 inch at a distance of 3 inch from origin. In sketch 3, two circles of diameters 0.545 inch and 0.625 inch are made concentric to previous circle. In sketch 4, two circles of diameters 0.625 inch and 0.785 inch are made concentric to previous circles. In sketch 5, two circles of diameters 0.785 inch and 0.875 inch are made concentric to previous circles.

3.1.2 Sweep

Sketch 2, 3, 4 & 5 are swept along the line made in sketch made in sketch 1 using the" add frozen" operation to construct the 3D model with different parts. The helical sweep is of 2 turns because the twist specification is defined in number of turns.

3.1.3 Merging

After sweep operation, it will show the model as 4 parts and 4 bodies. For merge operation, all the 4 parts are selected using control and merged as 1 part. At the end it will show as 1 part and 4 bodies. The 4 bodies within 1 part are named as follows:

Table 1 Naming of various parts of the body with state type

Part number	Part Of The Model	State Type
1	Inner_Fluid	Fluid
2	Inner_Pipe	Solid
3	Outer_Fluid	Fluid
4	Outer_Pipe	Solid

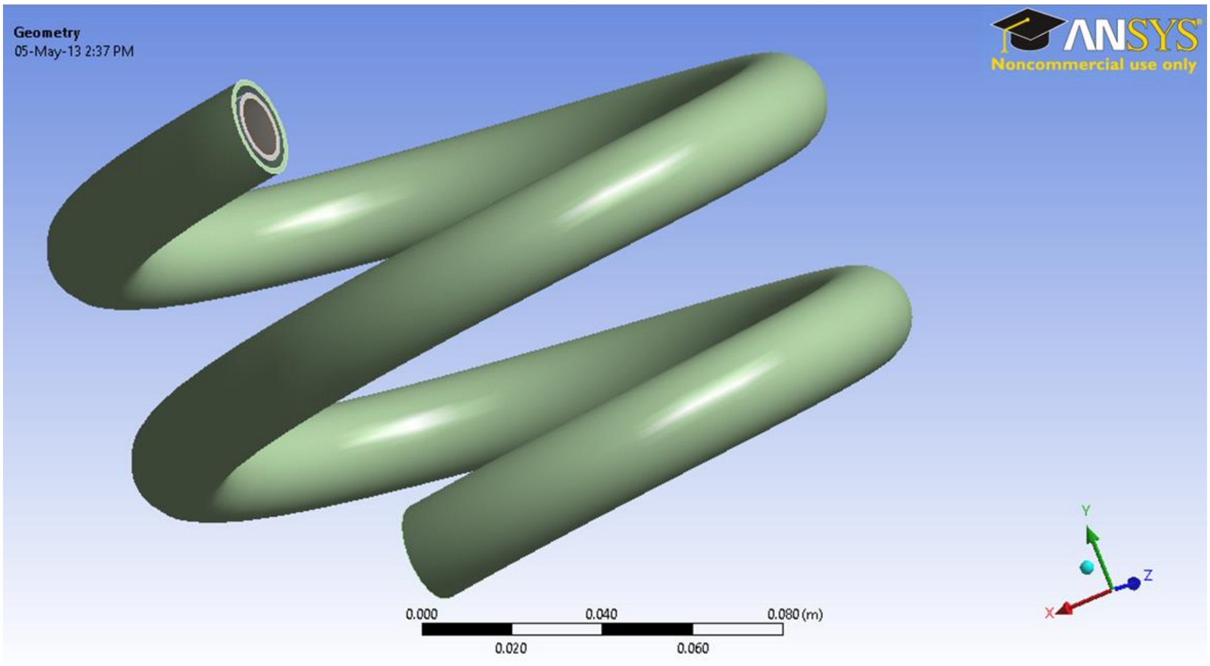


Figure 3 Original Geometry

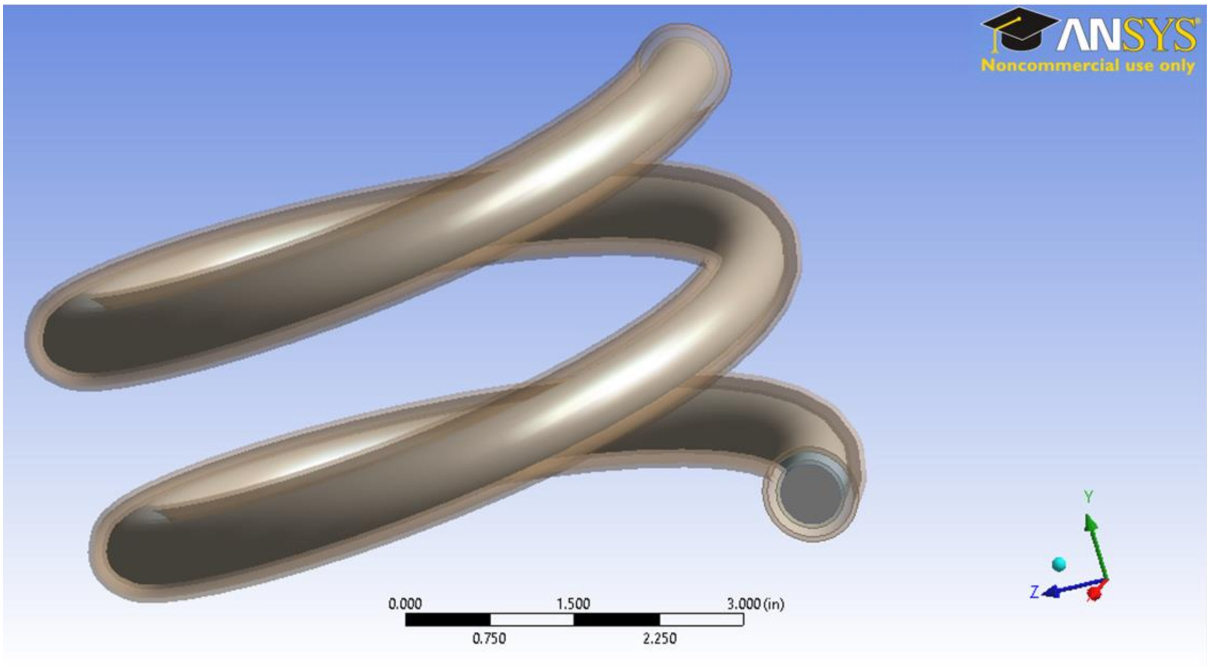


Figure 4 Geometry for Meshing

Save the project at this point and close the window. Refresh and update the project on the workbench.

3.2 Mesh:

Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

3.2.1 y^+ Values

y^+ values play a significant role in turbulence modeling for the near wall treatment. y^+ is a non-dimensional distance. It is often used to describe how coarse or fine a mesh is for a particular flow pattern. It determines the proper size of the cells near domain walls. The turbulence model wall laws have restrictions on the y^+ value at the wall. For instance, the standard K-epsilon model requires a wall y^+ value between approximately 300 and 100. A faster flow near the wall will produce higher values of y^+, so the grid size near the wall must be reduced. y^+ values for different wall treatments are given in table 2

Table 2 y+ Values for Different Wall Treatments

Wall treatment method	Recommended y^+ values	Used y+ values at tube walls
Standard wall functions	$30 < y^+ < 400$	$y^+ < 5$
Non-equilibrium wall functions	$30 < y^+ < 100$	$y^+ < 5$
Low Reynolds number model	$y^+ \cong 1$	$y^+ < 1$

The mesh details view gave us the following information:

- Relevance centre: fine meshing
- Smoothing: high
- Size: 4.033e-005m to 8.066e-005m
- Pinch tolerance: 3.6297e-005m
- Nodes: 586300
- Elements: 53170

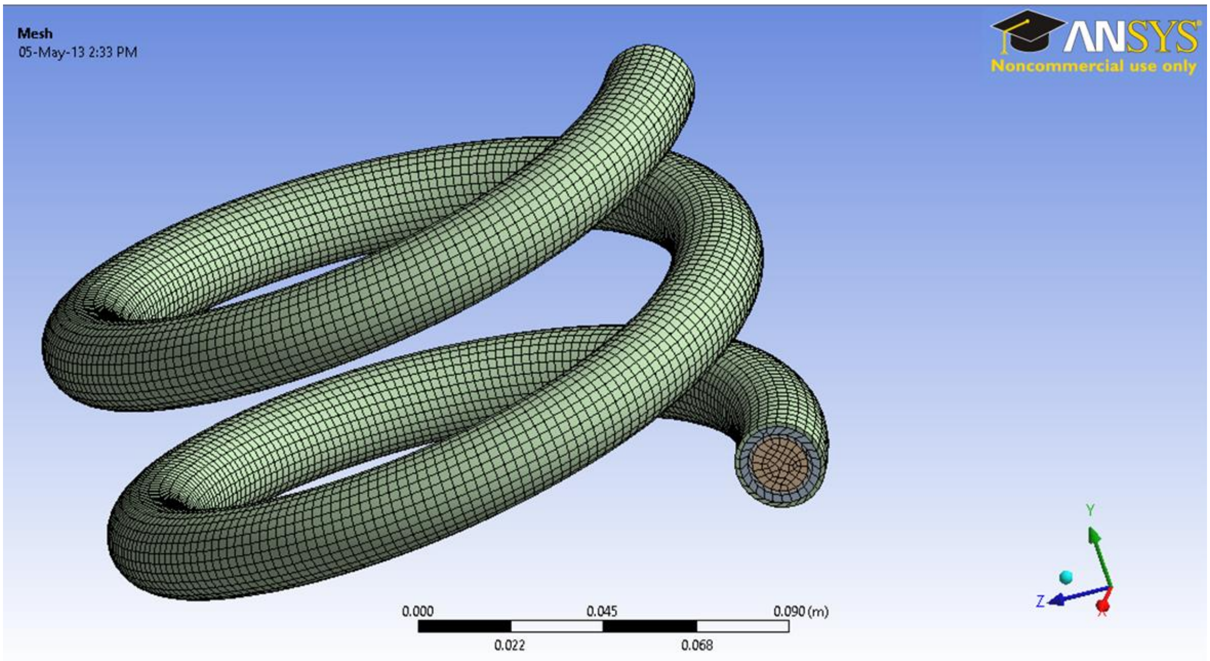


Figure 5 Mesh

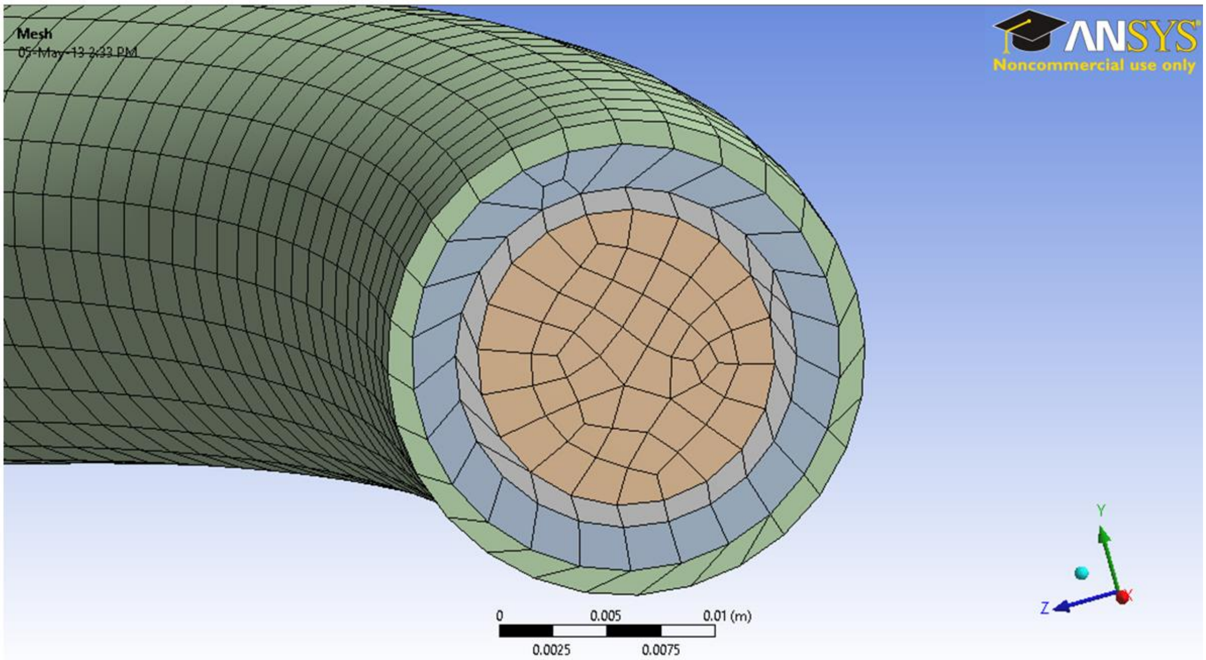


Figure 6 Close-up View of the Mesh

3.2.2 Named Selection

The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids. The outer wall is named as insulation surface.

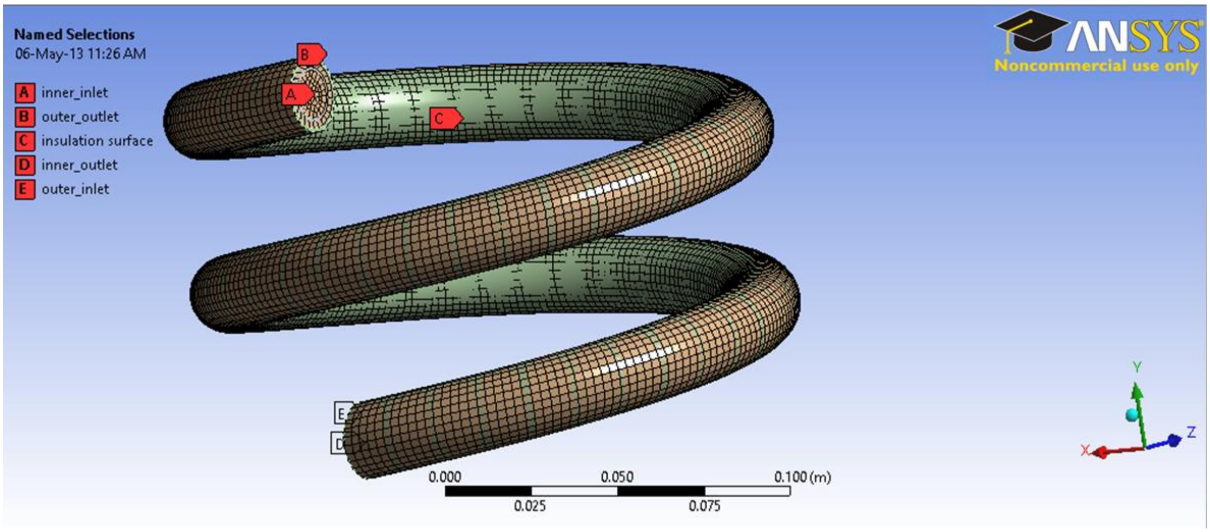


Figure 7 Named Selections

Save project again at this point and close the window. Refresh and update project on the workbench. Now open the setup. The ANSYS Fluent Launcher will open in a window. Set dimension as 3D, option as Double Precision, processing as Serial type and hit OK. The Fluent window will open.

3.3 Solution:

3.3.1 Problem Setup

The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state. Gravity is defined as y = -9.81 m/s^2

3.3.2 Models

Energy is set to ON position. Viscous model is selected as "k-ε model (2 equations). Radiation model is changed to Discrete Ordinates.

3.3.3 Materials

The create/edit option is clicked to add water-liquid and copper to the list of fluid and solid respectively from the fluent database.

3.3.4 Cell zone conditions

The parts are assigned as water and copper as per fluid/solid parts.

3.3.5 Boundary Conditions

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table 3 as given below.

Table 3 Boundary Conditions

-	Boundary Condition Type	Velocity Magnitude	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Temperature
Inner_Inlet	Velocity Inlet	0.9942 m/s	0.01 $m^2/^{s2}$	0.1 m^2/s^3	348 K
Inner_Outlet	Pressure Outlet	-	-	-	-
Outer_Inlet	Velocity Inlet	1.8842 m/s	0.01 m^2/s^2	0.1 m^2/s^3	283 K
Outer_Outlet	Pressure Outlet	-	-	-	-

3.3.6 Reference Values

The inner_inlet is selected from the drop down list of “compute from”. The values are:

- Area = 1 m^2
- Density = 998.2 kg/m^3
- Length = 39.37008 inch
- Temperature = 348 K
- Velocity = 0.9942 m/s
- Viscosity = 0.001003 kg/m-s
- Ratio of specific heats = 1.4

3.3.7 Solution Methods

The solution methods are specified as follows:

- Scheme = Simple

- Gradient = Least Square Cell Based
- Pressure = Standard
- Momentum = Second Order Upwind
- Turbulent Kinetic Energy = Second Order Upwind
- Turbulent Dissipation Rate = Second Order Upwind

3.3.8 Solution Control and Initialization

Under relaxation factors the parameters are

- Pressure = 0.3 Pascal
- Density = 1 kg/m^3
- Body forces = 1 kg/m^2s^2
- Momentum = 0.7 kg-m/s
- Turbulent kinetic energy = 0.8 m^2/s^2

Then the solution initialization method is set to Standard Initialization whereas the reference frame is set to Relative cell zone. The inner_inlet is selected from the compute from drop down list and the solution is initialized.

3.3.9 Measure of Convergence

It is tried to have a nice convergence throughout the simulation and hence criteria is made strict so as to get an accurate result. For this reason residuals are given as per the table 4 that follows.

Table 4 Residuals

Variable	Residual
x-velocity	10^{-6}
y-velocity	10^{-6}
z-velocity	10^{-6}
Continuity	10^{-6}
Specific dissipation energy/ dissipation energy	10^{-5}
Turbulent kinetic energy	10^{-5}
Energy	10^{-9}

3.3.10 Run Calculation

The number of iteration is set to 500 and the solution is calculated and various contours, vectors and plots are obtained.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Mass Flow Rate and Total Heat Transfer Rate

The mass flow rate and total heat transfer rate are given in the tables below.

Table 5 Mass Flow Rate

Mass Flow Rate	(kg/s)
inner_inlet	0.14690745
inner_outlet	-0.14690745
interior-part-inner_fluid	60.470463
interior-part-outer_fluid	-83.482896
outer_inlet	0.21353984
outer_outlet	-0.21353761
Net	2.2285868e-06

Table 6 Total Heat Transfer Rate

Total Heat Transfer Rate	(w)
inner_inlet	30626.679
inner_outlet	-23526.716
outer_inlet	-13529.476
outer_outlet	6429.4192
Net	-0.094444952

4.2 Contours

The temperature, pressure and velocity distribution along the heat exchanger can be seen through the contours.

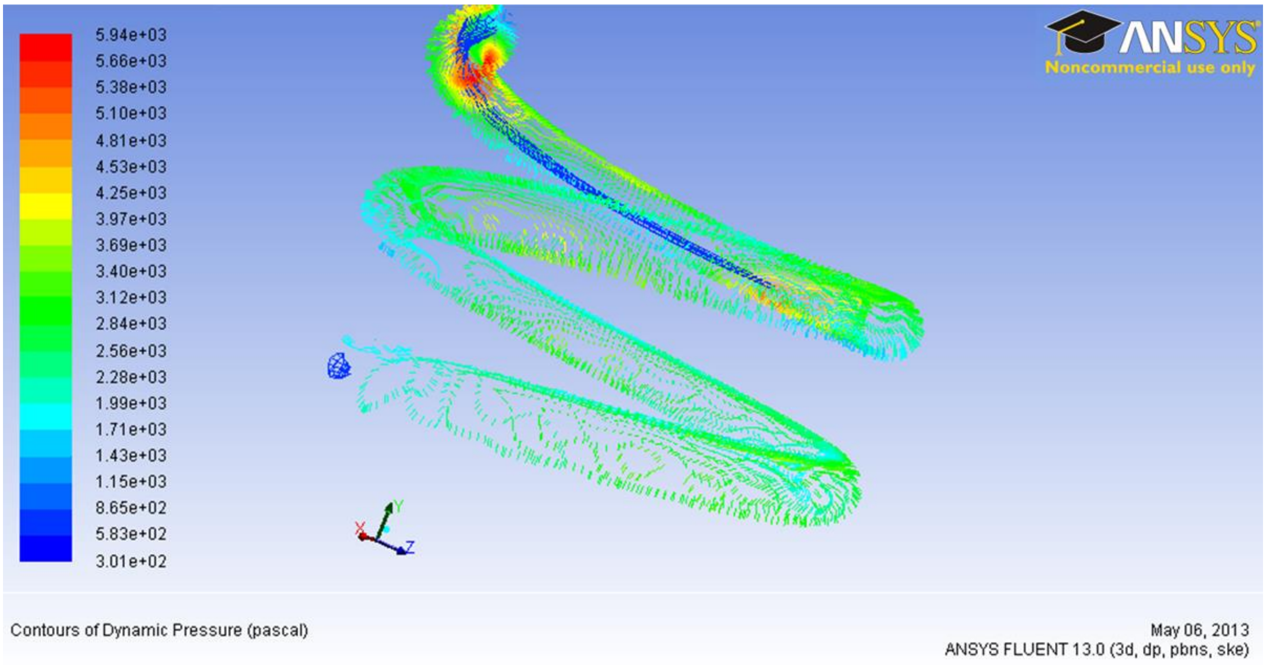


Figure 8 Contours of Dynamic Pressure in Pascal

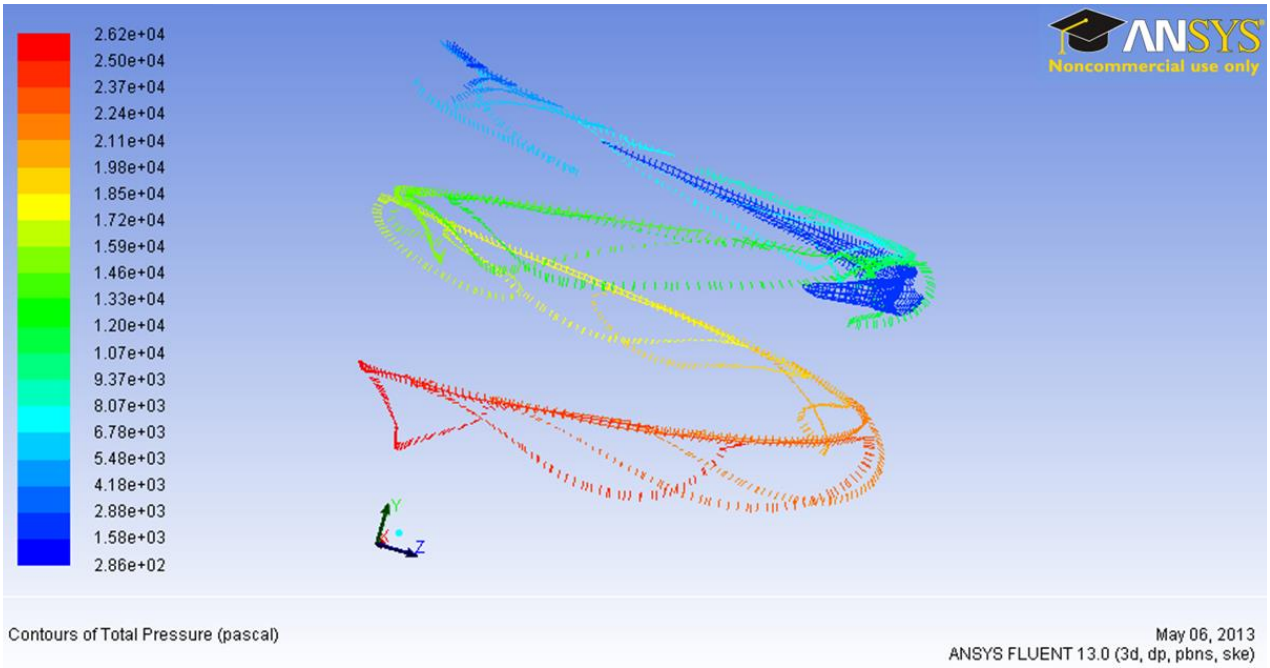


Figure 9 Contours of Total Pressure in Pascal

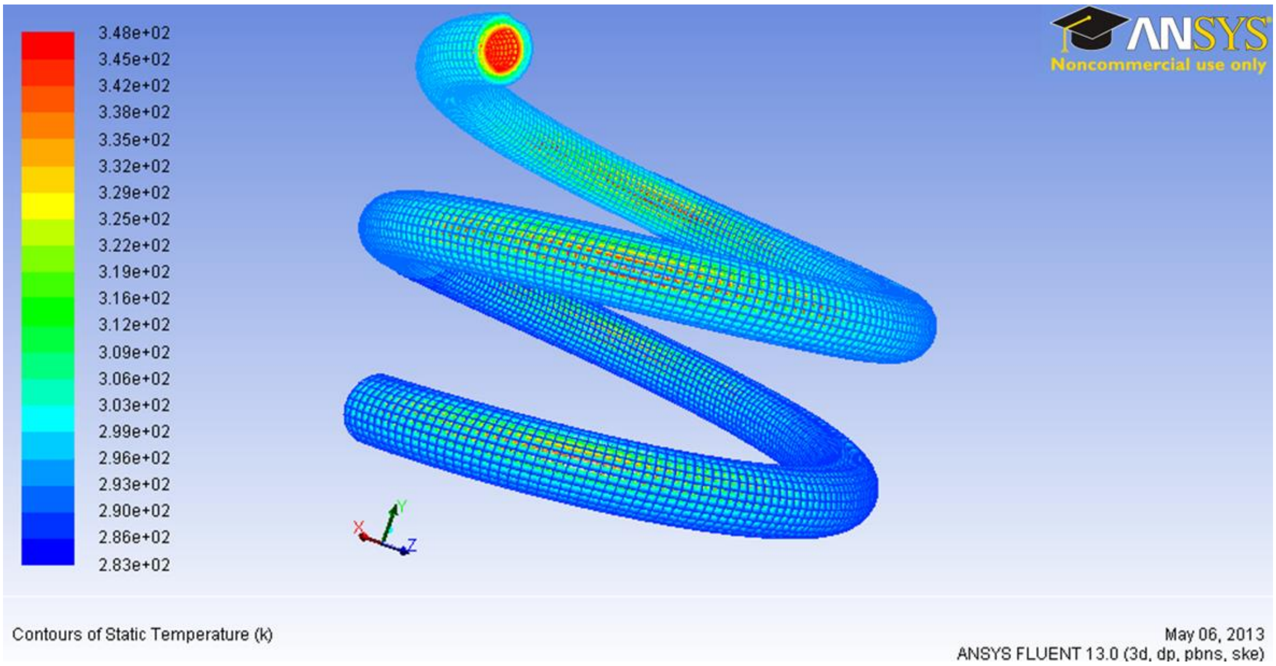


Figure 10 Contours of Static Temperature in K

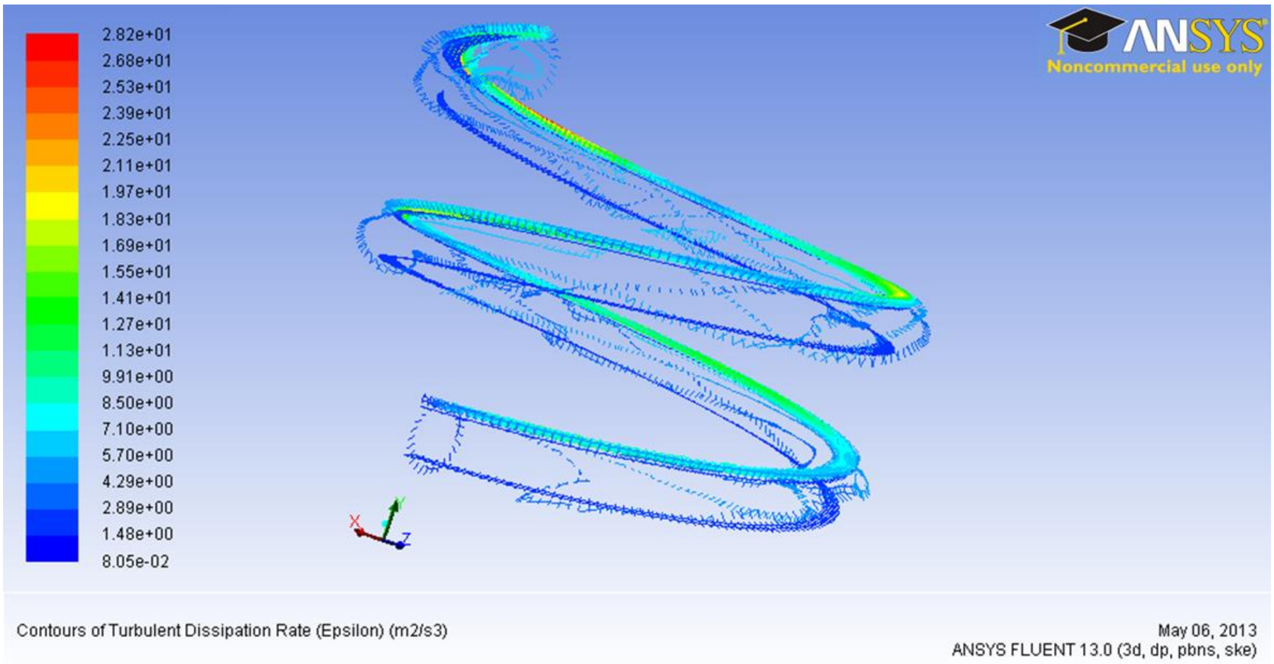


Figure 11 Contours of Turbulent Dissipation Rate (Epsilon) (m^2/s^3)

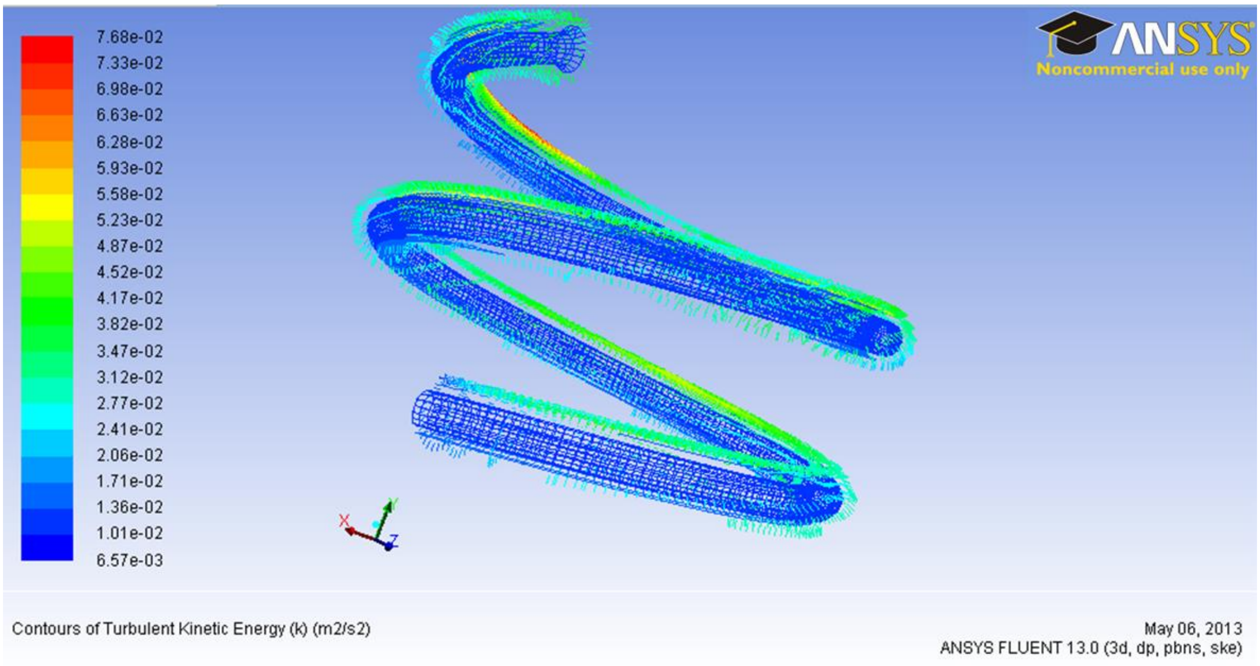


Figure 12 Contours of Turbulent Kinetic Energy k (m^2/s^2)

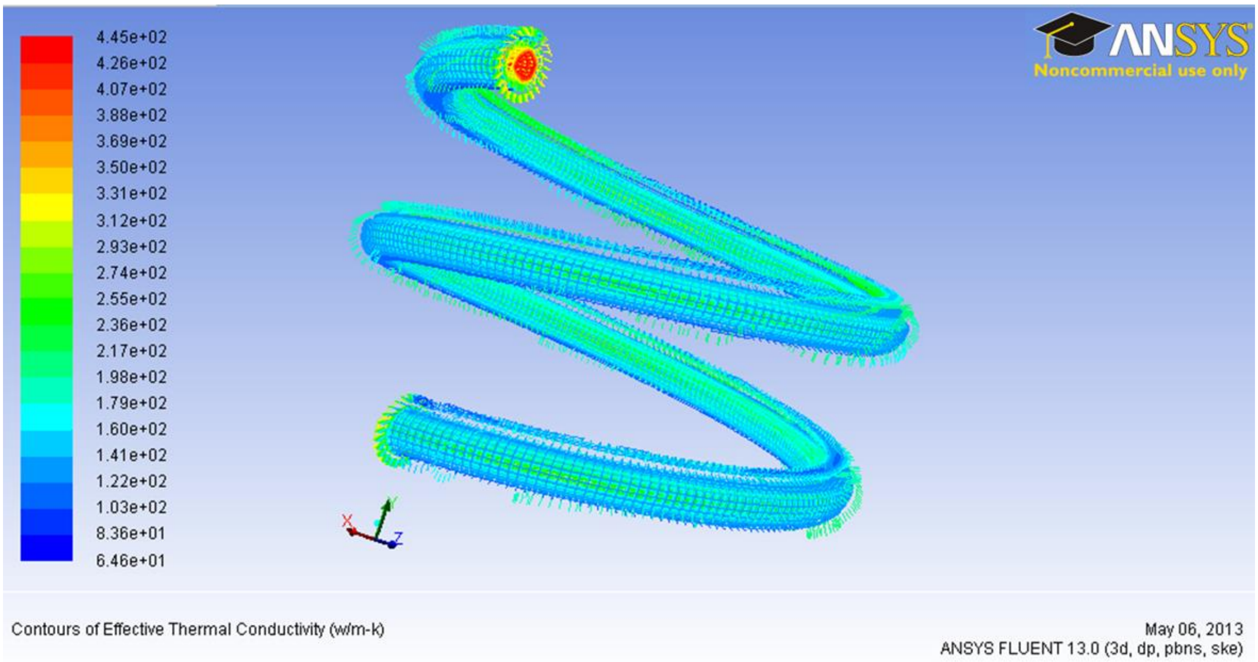


Figure 13 Contours of Effective Thermal Conductivity (w/m-K)

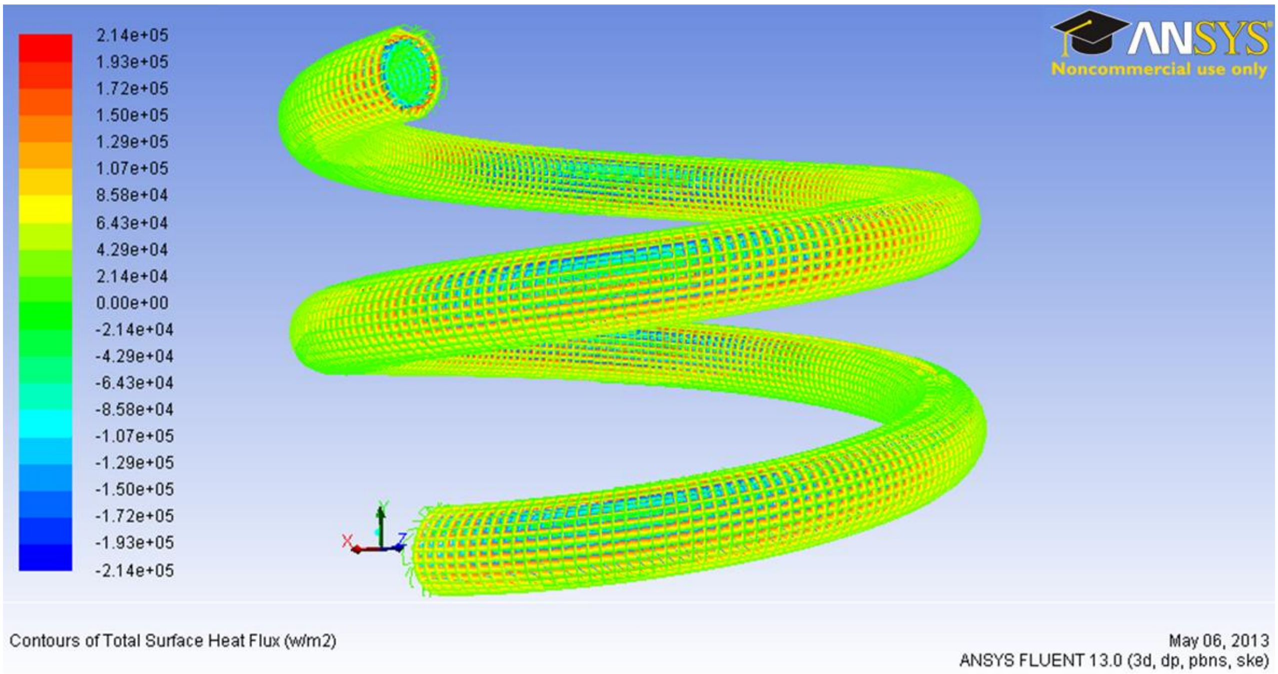


Figure 14 Contours of Total Surface Heat Flux (w/m^2)

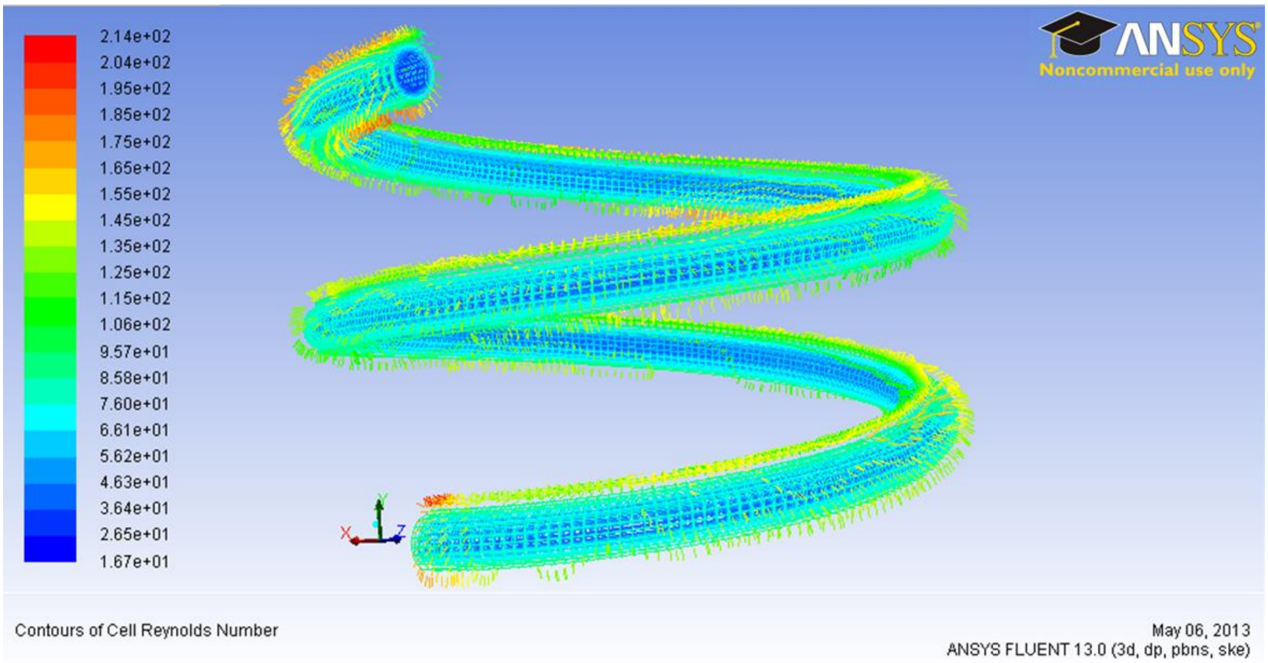


Figure 15 Contours of Cell Reynolds Number

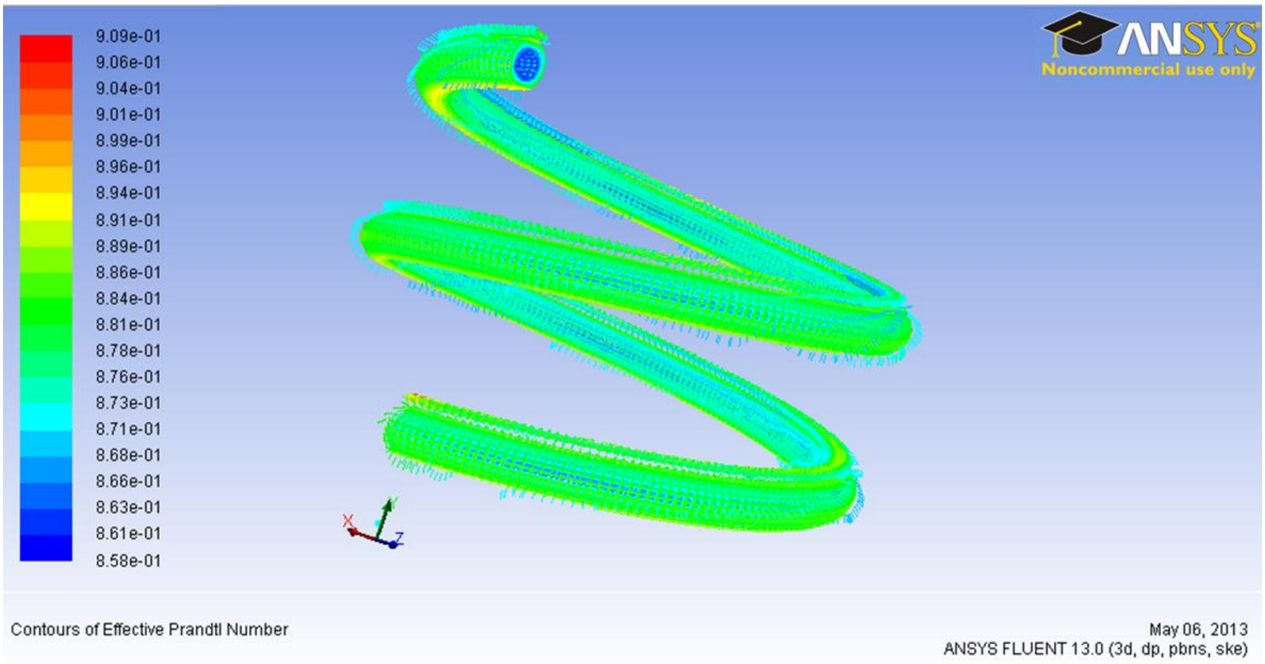


Figure 16 Contours of Effective Prandtl Number

4.3 Vectors

The plots give an idea of flow separation at several parts of the heat exchanger.

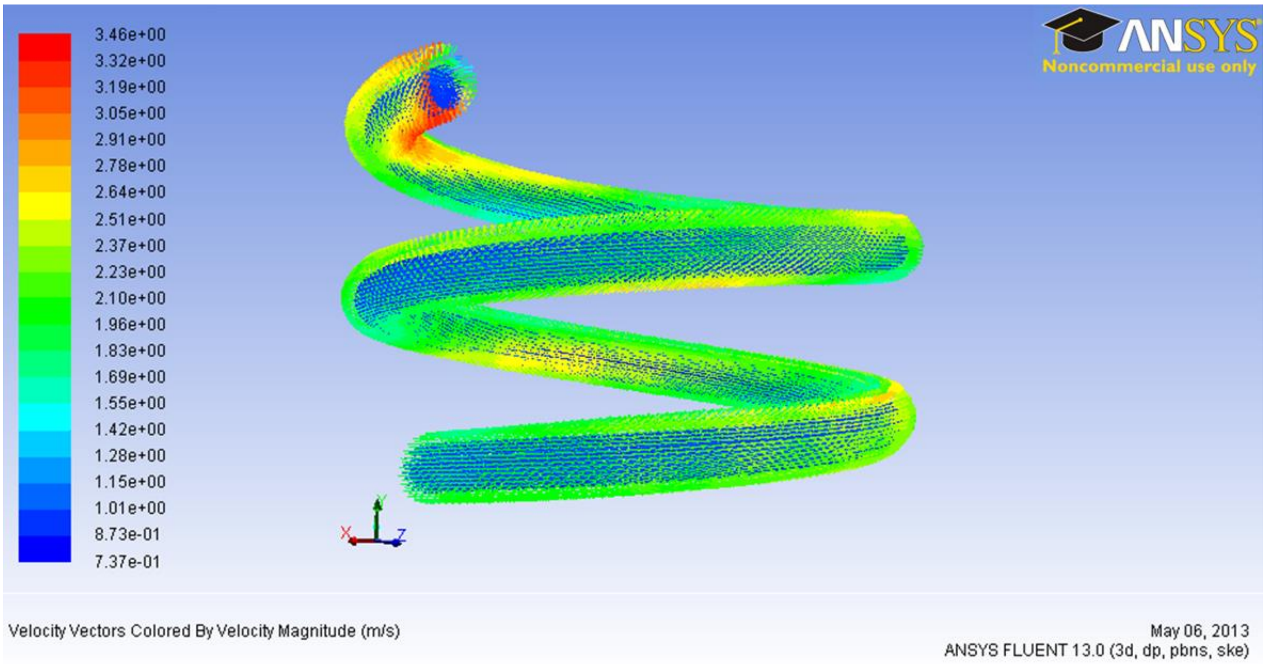


Figure 17 Velocity Vectors Colored by Velocity Magnitude (m/s)

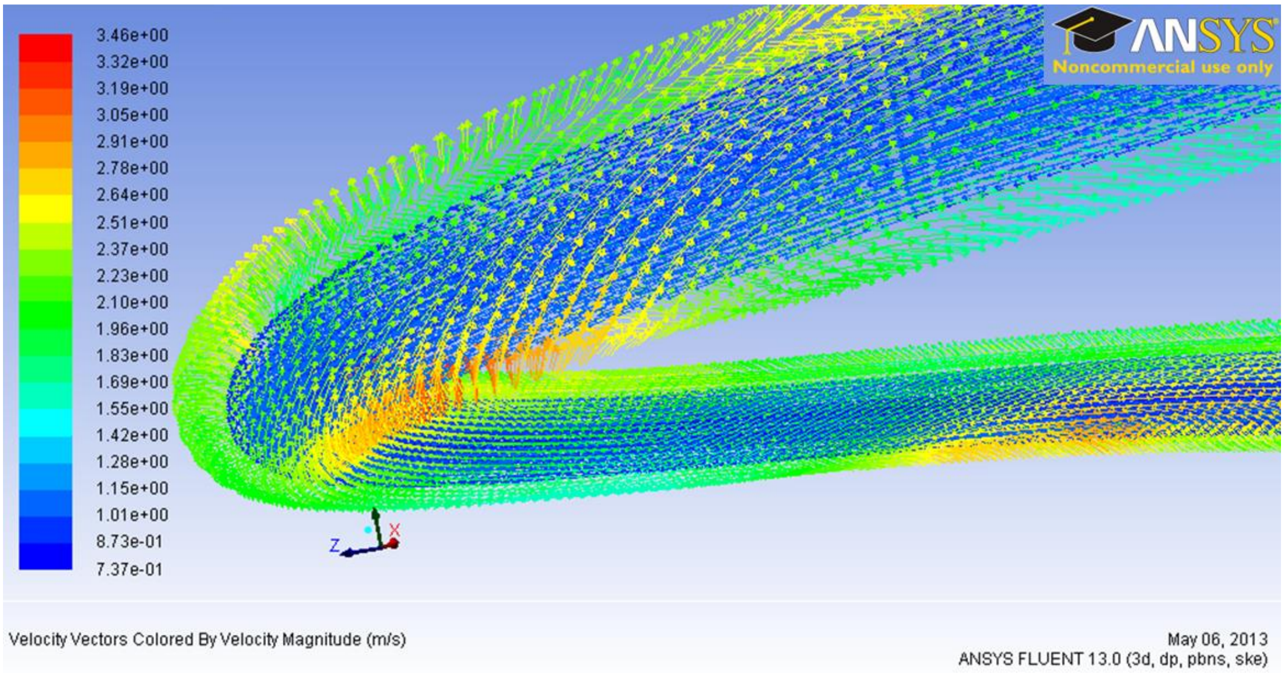


Figure 18 Close View of Velocity Vectors Colored by Velocity Magnitude (m/s)

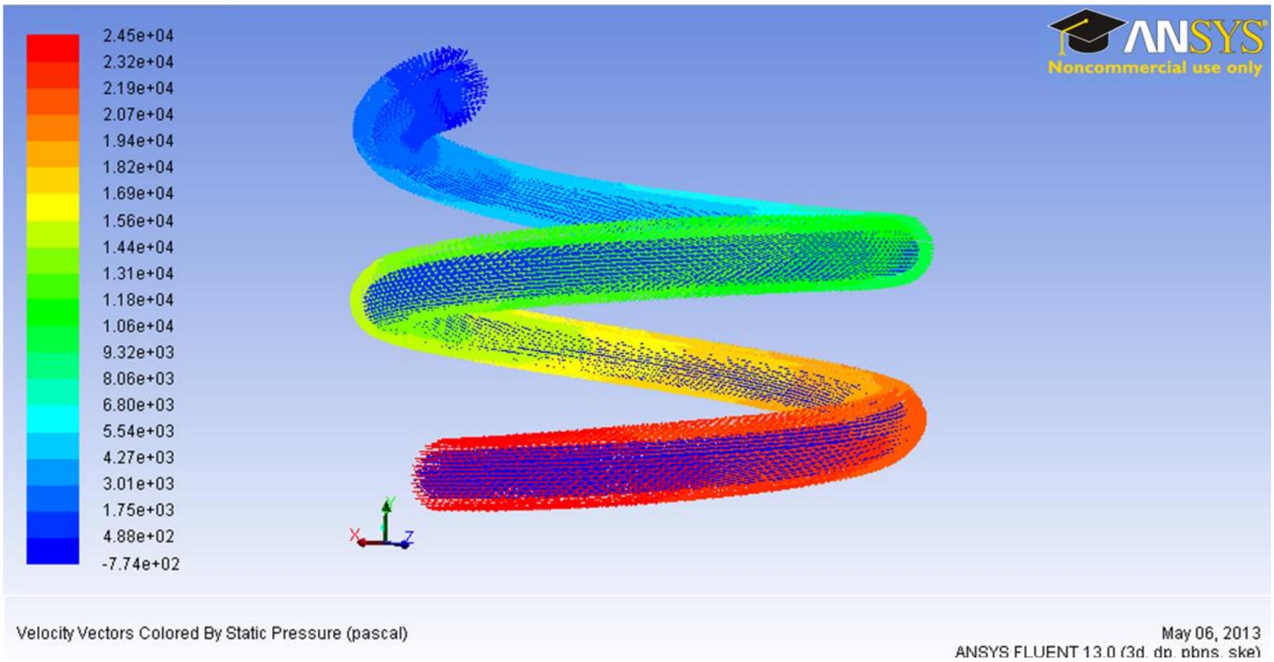


Figure 19 Velocity Vectors Colored by Static Pressure (Pascal)

4.4 Plots

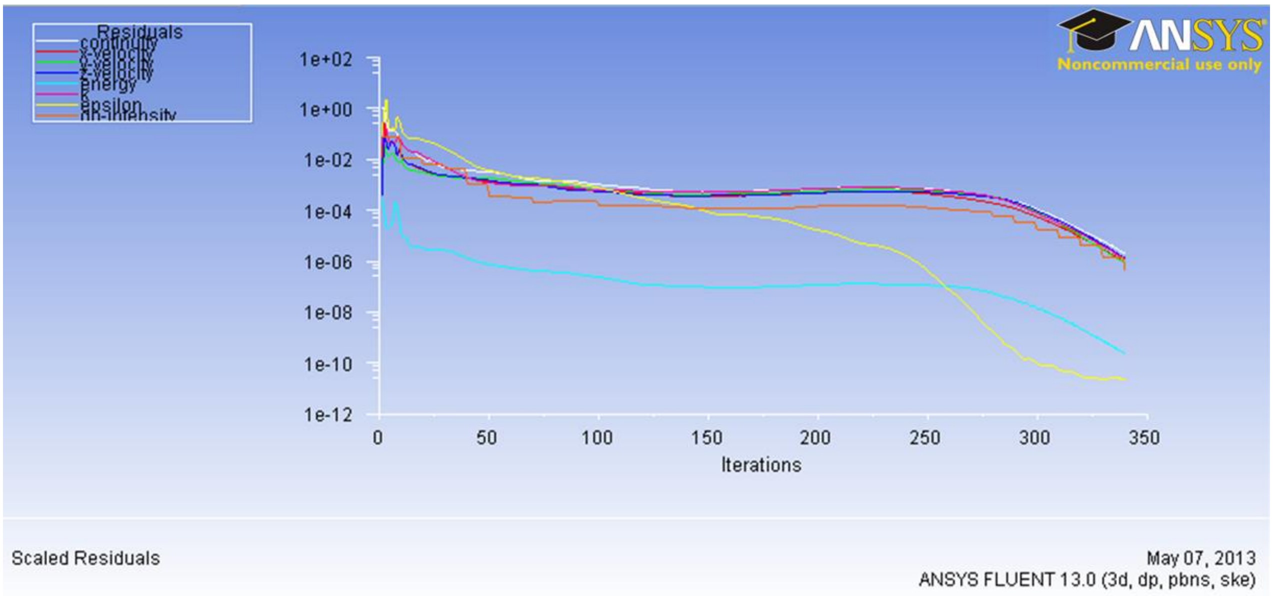


Figure 20 Scaled Residuals

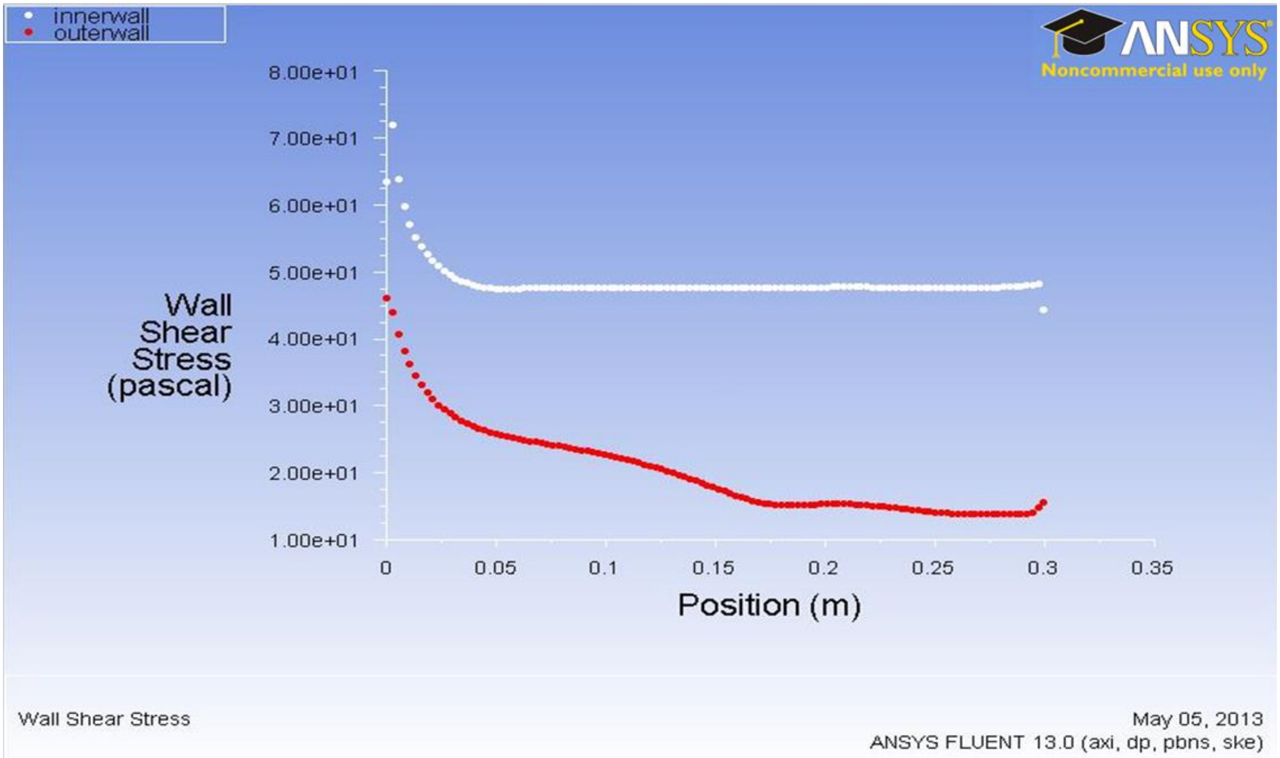


Figure 21 Wall Shear Stress Plot for Inner-wall And Outer-wall

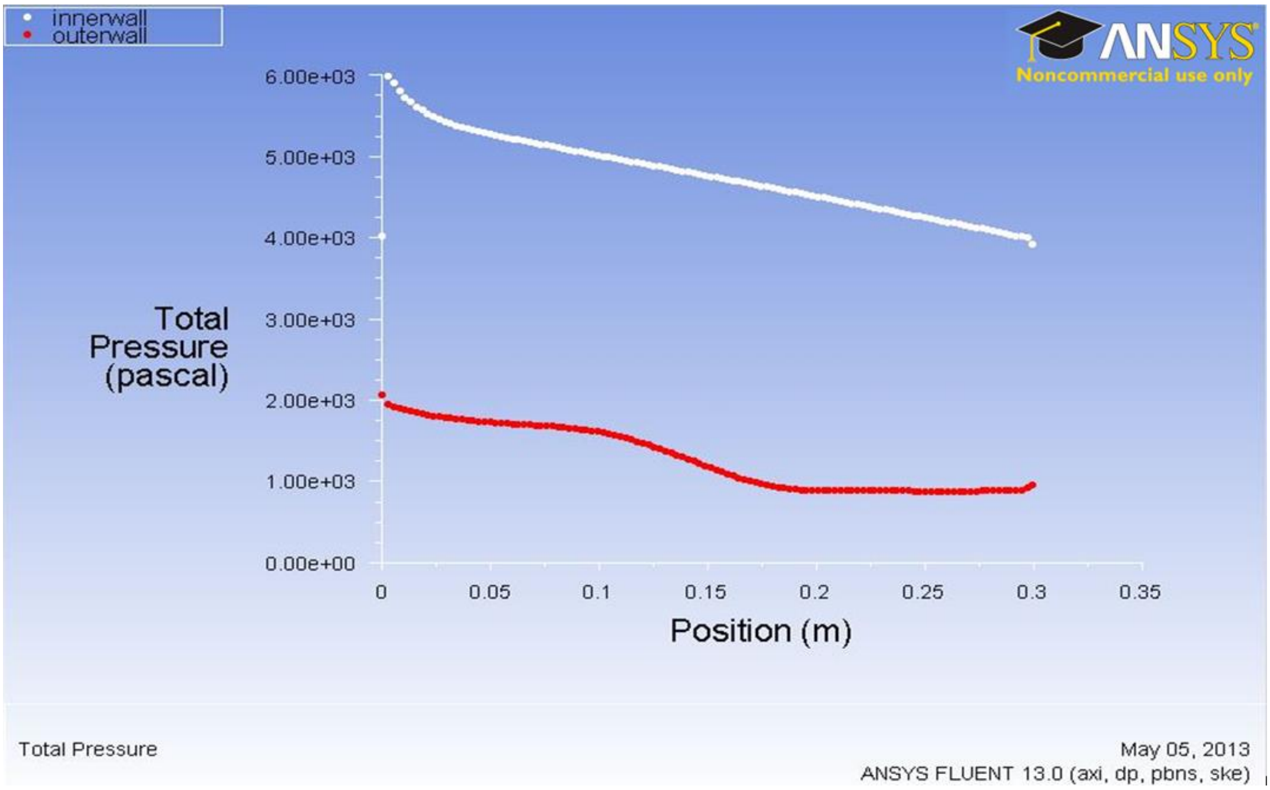


Figure 22 Total Pressure Plot for Inner-wall And Outer-wall

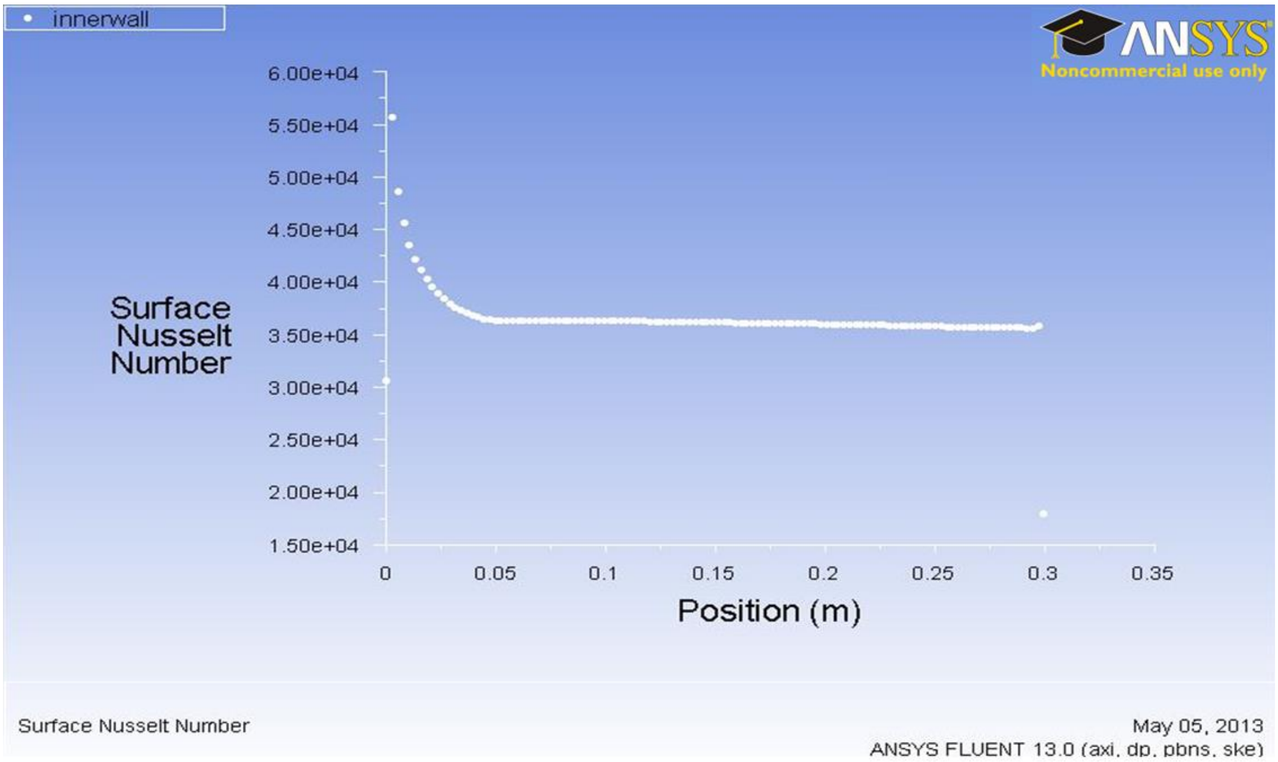


Figure 23 Surface Nusselt Number Plot for Inner-wall

CONCLUSIONS

A CFD package (ANSYS FLUENT 13.0) was used for the numerical study of heat transfer characteristics of a helical coiled double pipe heat exchanger for counter flow and the results were then compared with that of the parallel flow. The CFD results when compared with the experimental results from different studies were well within the error limits. The study showed that there is not much difference in the heat transfer performances of the parallel-flow configuration and the counter-flow configuration. Nusselt number at different points along the pipe length was determined from the numerical data. The simulation was carried out for water to water heat transfer characteristics and different inlet temperatures were studied. Nusselt number for the pipes was found to be varying from 340-360.

Characteristics of the fluid flow were also studied for the constant temperature and constant wall heat flux conditions. From the velocity vector plot it was found that the fluid particles were undergoing an oscillatory motion inside both the pipes.

From the pressure and temperature contours it was found that along the outer side of the pipes the velocity and pressure values were higher in comparison to the inner values.

REFERENCES

1. Experimental and CFD study of a single phase cone-shaped helical coiled heat exchanger: an empirical correlation. By Daniel Flórez-Orrego, ECOS June 26-29, 2012.
2. Helically Coiled Heat Exchangers by J.S.Jayakumar.
3. Numerical And Experimental Studies of a Double pipe Helical Heat Exchanger by Timothy John Rennie, Dept. of Bio-resource Engg. McGill University, Montreal August 2004.
4. Experimental and CFD estimation of heat transfer in helically coiled heat exchangers by J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, and Rohidas Bhoi, 2008, Chemical Engg Research and Design 221-232.
5. Heat Transfer Optimization of Shell-and-Tube Heat Exchanger through CFD Studies by Usman Ur Rehman, 2011, Chalmers University of Technology.
6. Structural and Thermal Analysis of Heat Exchanger with Tubes of Elliptical Shape by Nawras H. Mostafa Qusay R. Al-Hagag, IASJ, 2012,Vol-8 Issue-3.
7. Numerical analysis of forced convection heat transfer through helical channels Dr. K. E. Reby Roy, IJEST, July-2012 vol-4.
8. Minton P.E., Designing Spiral Tube Heat Exchangers, Chemical Engineering, May 1970, p. 145.
9. Noble, M.A., Kamlani, J.S., and McKetta, J.J., Heat Transfer in Spiral Coils, Petroleum Engineer, April 1952, p. 723.

Heat Transfer Analysis of Helical Coil Heat Exchanger with Circular and Square Coiled Pattern by Ashok B. Korane, P.S. Purandare, K.V. Mali, IJESR, June 2012, vol-2, issue-6.